AE451 - Experiments in Aerospace Engineering-III.

Laminar and turbulent boundary layers characteristics on a flat plate

Learning objective:

To understand the features of laminar and turbulent boundary layers and characterize it based on its velocity profile, displacement thickness, momentum thickness, using a Pitot tube and a differential manometer.

Proposed Plan:

a) Make a sketch of the experimental set up. Note the geometry of the flat plate leading edge, and the smoothness of the surface.

b) Make sketch of the traversing mechanism and the miniature Pitot probe.

c) Make a block diagram of the experimental set up including the DAQ system.

d) Mount the miniature Pitot tube in the horizontal traversing mechanism. Connect the Pitot tube lead to a micro-manometer.

e) At one free stream wind speed at a stream-wise station, x, measure velocity profiles in the laminar and the turbulent boundary layer. The origin of x coordinate is at the flat plate leading edge and it is increasing in the stream-wise direction. Note that you can have turbulent boundary layer at the same stream-wise station by introducing roughness or a tripping element at the leading edge of the plate.

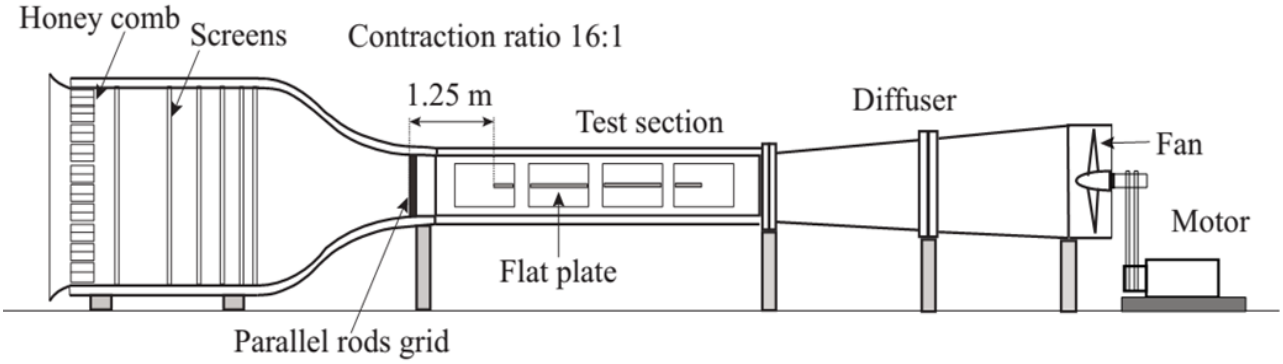
f) Plot velocity profiles, and obtain wall shear stress and boundary layer thicknesses. Note the differences between laminar and turbulent boundary layer velocity profiles and understand why they are different.

Equipment's used:

- Low speed wind tunnel
- Flat plate
- Pitot tube
- Digital differential manometer
- NI data acquisition card.
- Computer

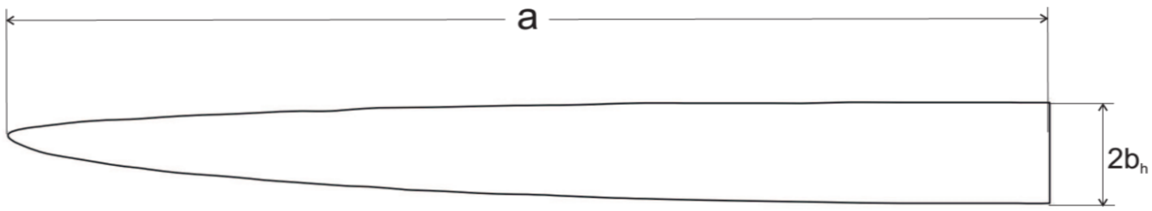
Wind Tunnel specifications:

- Open non-return suction type
- Contraction ratio : 16:1
- Operational speed : 3 to 18 m/s.
- Test section : 610 x 610mm square cross section, 10ft long.



- No of screens : 6, square mesh 1mm.
- Honeycomb : square 1ft long.
- Motor Specs : A.C induction motor, 15hp, 50 Hz, 1445 max rpm.
- Diffuser : Square section of 10ft long.

Flat plate Specifications:



- Leading edge - Asymmetrical Modified super ellipse
- Material - Acrylic-glass
- Geometry - 2 m long, 0.61 m width, 12 mm thick
- Semi major axis, a =120mm; AR_u= 30, AR_l = 15.

Pitot tube:

- Used for measuring the total pressure of the flow.
- Static pressure is measured using a port drilled in the tunnel wall.

Digital differential manometer:

- Range – 20 mm of H_2O (= 200 Pascal which is = 5V).
- Furness FC012 digital manometer.

Data Acquisition System:

- PXI – 1073 chassis.
- NI-PXI 6251, 16 bit data acquisition card.
- BNC 2120 signal I/O board

Questions:

1) What do you understand by the probe displacement effect?
2) How can you obtain local shear stress from the velocity profile data?
3) What are displacement thickness and momentum thickness? How are they related to one another in the case of laminar and turbulent boundary layers?
4) Maximum resolution of the DAQ system?
5) Why does a laminar boundary layer become turbulent?
6) What do you understand by the transition and intermittency in a flat plate boundary layer? What is the critical Reynolds number for transition over a flat plate?

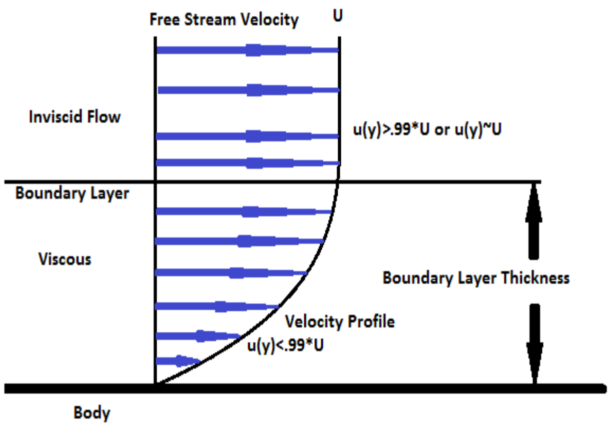
References:

1. Boundary layer theory - Schlichting
2. Turbulence - Hinze
3. A first course in turbulence - Tennekes & Lumley
4. Viscous fluid flow - F.M. White
5. Fluid mechanics - Kundu & Cohen
6. Fluid mechanics - F.M. White

Pre-requisite:

Boundary layer:

A thin layer close to the solid body where the fluid velocity increases from zero at the wall (due to no slip condition for viscous fluids) to its maximum value at the free stream(U_∞) is called the boundary layer. The variations (i.e. velocity gradients) will be confined within this thin layer close to the wall as long as the Reynolds number is large enough or the fluid viscosity is small enough.



Usual thickness of BL (@ U =6 m/s)

δ (laminar) = **6 mm**

δ (turbulent) = **10 mm**

Features of boundary layer:

- Strong velocity gradient in the normal direction exists hence shear stresses are always present though the viscosity is small, because

$$\tau = \mu \frac{\partial u}{\partial y}, \quad where\ \frac{\partial u}{\partial y}\ is\ the\ velocity\ gradient$$

- **δ↑ as - U_∞↓, ν↑, x↑, Re↓**

Where **δ** is the BL thickness, **U_∞** is the free-stream velocity, **ν** is the kinematic viscosity, '**x**' is the characteristic length and **Re** is the Reynolds number.

Assumptions: (steady, Re >> 1, Incompressible, Newtonian, no body force)

- δ << L, BL thickness is very small compared with the characteristic length of the flow ('x' – distance from the leading edge for flat plate, D for cylinder and sphere, H/2 for channel flows, etc.)
- transverse velocity, **v** << longitudinal velocity, **u**

$$\frac{\partial u}{\partial x} \ll \frac{\partial u}{\partial y}, \quad \frac{\partial v}{\partial x} \ll \frac{\partial v}{\partial y}$$

Boundary layer equations:

Navier-Stokes equation completely describes the motion of a fluid element subjected to body and surface forces.

$$\varrho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=X-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)$$

$$\varrho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=Y-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$

$$\varrho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=Z-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)$$

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0\,.$$

For boundary layer, we were able to simplify this NS equations based on the assumptions that were put forwarded by Prandtl which were described above, and derive boundary layer equations given below.

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\varrho}\frac{\mathrm{d}p}{\mathrm{d}x}+\nu\frac{\partial^2 u}{\partial y^2}$$

This equation can be applied as such for a laminar flow but for a turbulent flow these instantaneous velocities have to be decomposed into mean and a fluctuating part and must be averaged over a long period of time. Doing so, we get the Reynolds Averaged Navier Stokes (RANS) equation shown below.

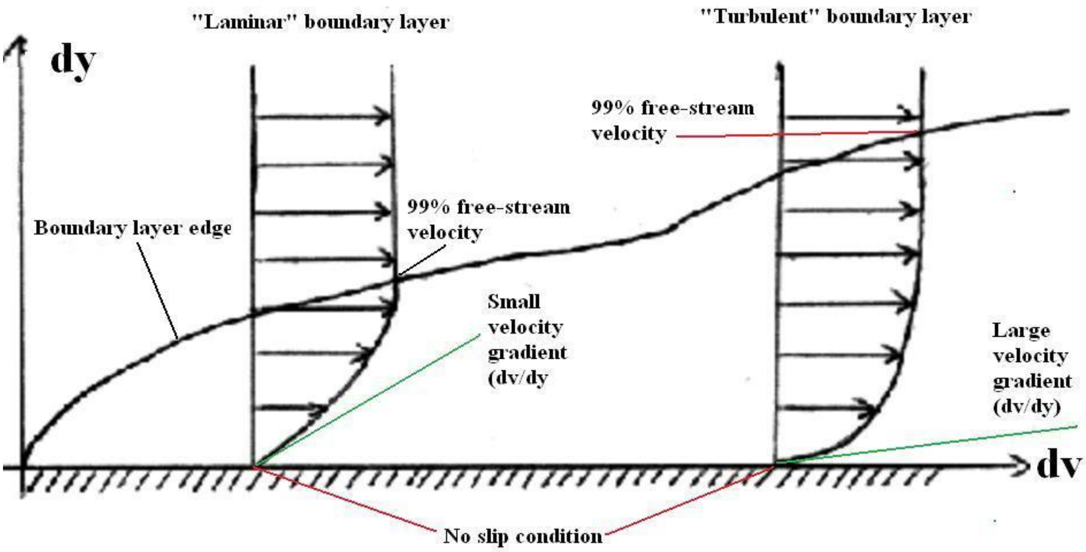
$$\frac{\partial \bar{u}}{\partial x}+\frac{\partial \bar{v}}{\partial y}=0\,,$$

$$\bar{u}\frac{\partial \bar{u}}{\partial x}+\bar{v}\frac{\partial \bar{u}}{\partial y}=-\frac{1}{\varrho}\frac{\partial \bar{p}}{\partial x}+\frac{\partial}{\partial y}\left\{\nu\frac{\partial \bar{u}}{\partial y}-\overline{u'\,v'}\right\}$$

For proper derivation of these equations one may refer to ‘Boundary Layer Theory’ by “Schlichting” and ‘Viscous fluid flow’ by “F. M. White”.

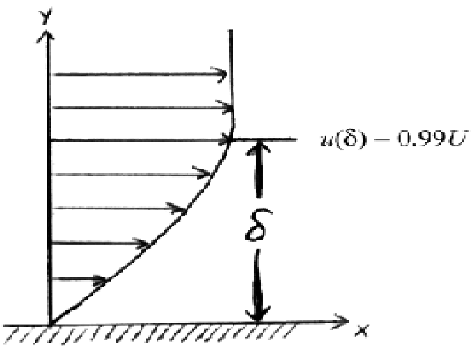
Laminar and turbulent boundary layer:

- A boundary layer can be laminar or turbulent depending on whether the flow inside the boundary layer is laminar or turbulent which in turn depends on the flow Reynolds number.
- Once the flow becomes turbulent many changes happens in the flow. Most of the properties like skin friction, heat transfer rate gets changed which in turn affects the drag experienced by a body, the amount of heat transfer from a hot body to a cold stream, efficiency of turbine blades, etc.
- Hence, a proper understanding of laminar and turbulent boundary layers is required, especially for aerospace engineers.



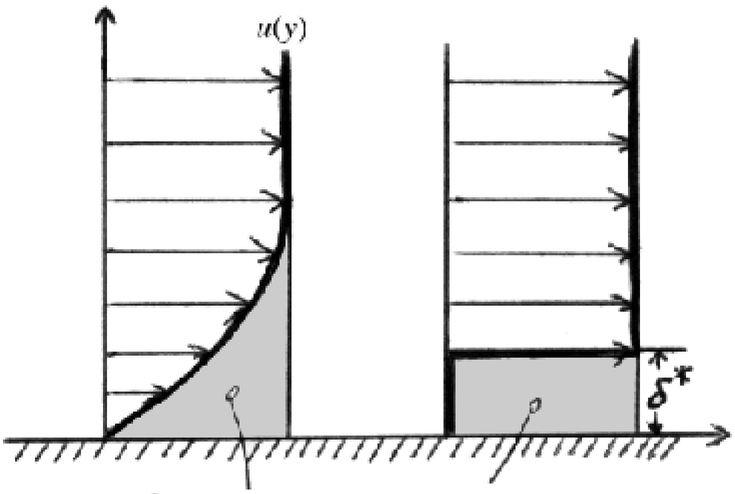
PARAMETERS:

Boundary layer thickness: The distance from the wall where the mean velocity reaches the 99% of free stream velocity.



$$\boldsymbol{\delta = y|_{@\, u=0.99*U_\infty}}$$

Displacement thickness: This is defined as the distance by which the wall would have to be displaced outward in a hypothetical frictionless flow so as to maintain the same mass flux as in the actual flow.



Shaded Areas Have Equal Area

$$\delta^* = \int_0^\infty \left(1 - \frac{u}{U}\right) dy$$

This is the distance by which the streamlines of the external flow are displaced because of the presence of the body.

Momentum thickness: Distance by which the streamlines have to be shifted in a potential flow, so that it produces the same momentum loss produced by the presence of boundary layer.

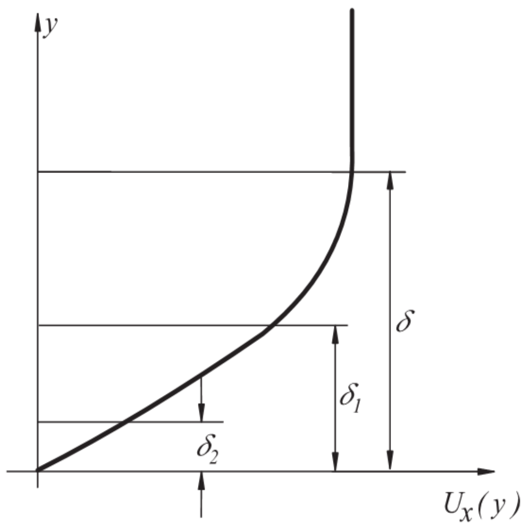
$$\theta = \int_0^\infty \frac{u}{U} \left(1 - \frac{u}{U}\right) dy$$

It can be visualized, by replacing u in the previous figure by ρu^2.

Shape factor: Ratio of displacement to momentum thickness, $H = \frac{\delta^*}{\theta}$, a parameter, which quantifies the relative importance of the mass and momentum diffusion.

All these parameters are common for both laminar and turbulent boundary layer but their values are different depending on the wall normal velocity profile.

Relative dimensions of these parameters:



How to calculate all these parameters experimentally?

- Obtain the velocity profile using the Pitot tube.
- Find the boundary layer thickness by using the formula, $\delta = y|_{@\ u=0.99*U_\infty}$
- Plot the profiles of $1 - \frac{u}{U_\infty}$ and $\frac{u}{U_\infty}\left\{1 - \frac{u}{U_\infty}\right\}$.
- Find the area under these two profiles. This gives displacement, δ* and momentum thickness,θ
- These steps are common for both laminar and turbulent boundary layer.
- The difference comes in the calculation of wall shear stress i.e. its friction coefficient.

Calculation of skin friction coefficient:

LAMINAR:

- Using analytical formula from Blasius solutions given by

$$C_f = \frac{0.664}{\sqrt{Re_x}}$$

- It can also be checked with our experimental results based on the formula, $C_f = \frac{\theta}{x}$.

TURBULENT:

- By fitting a curve between $\frac{\boldsymbol{u}}{\boldsymbol{U_\infty}}$ and $\mathbf{ln}\,\frac{\boldsymbol{yU_\infty}}{\boldsymbol{\vartheta}}$.

  The slope of the curve gives the value of C in equation given below. From C we can find C_f.

$$\frac{\overline{\boldsymbol{u}}}{\boldsymbol{U_\infty}} = \boldsymbol{C} * \mathbf{ln}\left(\frac{\boldsymbol{yU_\infty}}{\boldsymbol{\vartheta}}\right) + \boldsymbol{D}$$

$$\textbf{Where, } \boldsymbol{C} = \frac{\mathbf{1}}{\boldsymbol{K}}\sqrt{\frac{\boldsymbol{C_f}}{\mathbf{2}}}\ \boldsymbol{and}\ \boldsymbol{D} = \sqrt{\frac{\boldsymbol{C_f}}{\mathbf{2}}}\left\{\frac{\mathbf{1}}{\boldsymbol{K}}\boldsymbol{ln}\left\{\sqrt{\frac{\boldsymbol{C_f}}{\mathbf{2}}}\right\} + \boldsymbol{B}\right\}$$

With k=0.41 and B=5.0

- This has to be then compared with the analytical results obtained by assuming a power law relation for the velocity profile.

$$\boldsymbol{C_f} \approx \frac{\mathbf{0.027}}{(\boldsymbol{Re_x})^{1/7}}\ \textbf{, and}\ \boldsymbol{C_f} \approx \frac{\mathbf{0.020}}{(\boldsymbol{Re_\delta})^{1/6}}$$

Where Re_x, and Re_δ, are Reynolds number based on plate length and boundary layer thickness, respectively.

Experiment # 2: Part-A

Measure velocity profile, both mean and fluctuating component, in the turbulent wake behind a circular cylinder using hot-wire anemometer.

Learning Objectives:

a) Use of hot-wire anemometry for velocity measurement

b) Analysis of hot-wire data to obtain mean and fluctuating part of the velocities

c) Wake characteristics behind a circular cylinder

Proposed Plan:

a) Study the experimental set-up including each and every component with specification.

b) Study the circuit diagram for a constant temperature hot wire circuit.

c) Mount the hot-wire probe in the traversing mechanism and make the necessary signal connection.

d) Make a block diagram of the experimental set up.

e) Calibrate hot wire probe in the free stream without any model in the tunnel.

f) Use a rough cylinder model to generate turbulent wake

g) At one free stream wind speed at a stream-wise station, x, measure velocity profiles, both mean and fluctuating component, in the turbulent wake. The origin of x coordinate is at the center of the cylinder and it is increasing in the stream-wise direction.

h) Plot velocity profiles, both mean and rms, in normalized forms.

Questions:

1) How does constant temperature hot-wire anemometer work?

2) How do you calculate mean and rms velocity components from hot wire signal?

3) What is the nature of the hot-wire signal in the free stream?

4) What do you understand by the intermittency in a turbulent wake?

Experiment # 2: Part-B

Study the dependence of shedding frequency in the Karman vortex street of the circular cylinder wake on Reynolds number.

Learning Objectives:

To understand the principles of a Constant temperature anemometer and thereby to study the vortex shedding frequency of a cylinder wake and to observe the variation of Strohaul number with Reynolds number.

Proposed Plan:

a) You are provided with four circular cylinders of different diameters. Measure the diameters with the help of a digital Vernier.

b) Mount the hot wire in the near wake (about 3 diameters) of the circular cylinder. Turn the wind tunnel on. At an appropriate wind speed the hot wire signal will show a sine wave. Note the frequency of the sine wave. This corresponds to the frequency of the vortex shedding.

c) Increase the Reynolds number and note the variation in the shedding frequency.

d) Calculate Strouhal number, $S = fd/\nu$, and Reynolds number. Plot results as $S = f(\mathrm{Re})$.

Questions:

1) What do you observe when the hot wire probe is moved across the cylinder wake? Explain the reason.
2) Can you use this method to measure low wind speed? Explain.
3) Sketch the flow field on a circular cylinder as a function of Reynolds number. Start with the inviscid flow. Comment on drag and other characteristics.
4) What happens to the shedding frequency when the hot wire probe is moved across the cylinder near wake? Explain.

References:

1. Schlichting, ``Boundary layer theory," McGraw Hill, New York, 1970.
2. Roshko, ``On the development of turbulent wakes from vortex streets," NASA technical note 2913, March, 1953.
3. Turbulence - Hinze
4. A first course in turbulence - Tennekes & Lumley
5. L. Ong and J. Wallace, “The velocity field of the turbulent very near wake of a cylinder,” Experiments in Fluids, 20 (1966).

Details of experiment:

Experiment No. 3

Aerodynamic forces and moments on a generic aircraft model

Learning Objectives:

- Use of 3D section of the 5-D wind tunnel
- Six-component Balance
- Model attitude control system
- Data acquisition system
- Estimation of aerodynamic forces acting on the model under test. The analysis involves calculation of forces and moments.
- Force data acquisition is done in order to estimate the aerodynamic coefficients for performance evaluation of the model under test.Force data is acquired using six component internal balance and is analyzed. Sign convention for forces and moments play an important role in balance data acquisition during the test and data analysis later. Balance data analysis includes forces and moments calculation in various axes system.

Apparatus:

- Six component force balance:
- Whetstone bridge card (SCXI-1000)
- Data acquisition card (PXI-1033)
- Labview software:
- Aircraft model is used to do experiment.

Precautions

1. Check the individual resistance of the bridges.

2. Check the bridge for shorts with the grounds. This should show a very high resistance of the order of mega ohms

3. DO NOT SHORT THE OUTPUT LEADS ONCE THE OOWER SUPPLY IN "ON"

4. Check the direction of the outputs by applying small loads

5. Allow the bridges to warms up for at least one hour actual measurement.

6. Check the Balance voltage which should not be exceed limit of 6-component strain gage balance.

Principle of operation and Procedure

Our aircraft modelis mounted on the balance accurately. So it transmits the load to the balance during experiment. Balance is attached to the mechanism which provides pitching, yawing and rolling moment to the aircraft model. We are using **Six component force balance which** is a cylindrical body on which an aerodynamics model can be mounted to measure different forces acting on it by aerodynamic loading.**Whetstone bridge card (SCXI-1000**) is a constant DC voltage supply is required to activate the bridges. For this a SCXI-1314 NI card is used which facilitates DC voltage supply. **Data acquisition card (PXI-1033) whichis** the voltage output of the strain gauges is required to be measured under the loading on the balance. For this a data acquisition card PXI-1033 with maximum of 16 voltage input channels is used. Also Labview software facilitated to provide/acquire the input/output voltage signals to/from the strain gauge bridges using a specially built VI program.Usually, non-dimensionalized raw signal (mV/V_{Ex}) has been saved during the test and analyzed later. A linear least square fit for the variation of the normalized outputs with respects to a particulars load components is obtained.

Our step for doing the experiment

- Setting the model on six component balance in wind tunnel
- No wind experiment when no air flow over aircraft model
- Wind test experiment when air flowing over aircraft model in wind tunnel
- calculation of aerodynamic coefficients in body as well as in wind axis

Sign Convention

In general, right hand rule or Euler's convention is followed for all the axes system. Direction of positive forces and moments in body and wind axes are shown in the figure2.

Axes:

x-axis	:	positive forward,
y-axis	:	positive to starboard
z- axis	:	positive downward

Forces in model axes:

Axial force	:	A
Side force	:	Y
Normal force	:	N

Moments in model axes:

Rolling moment, l	:	starboard wing down
Pitching moment, m	:	nose up
Yawing moment, n	:	nose towards starboard

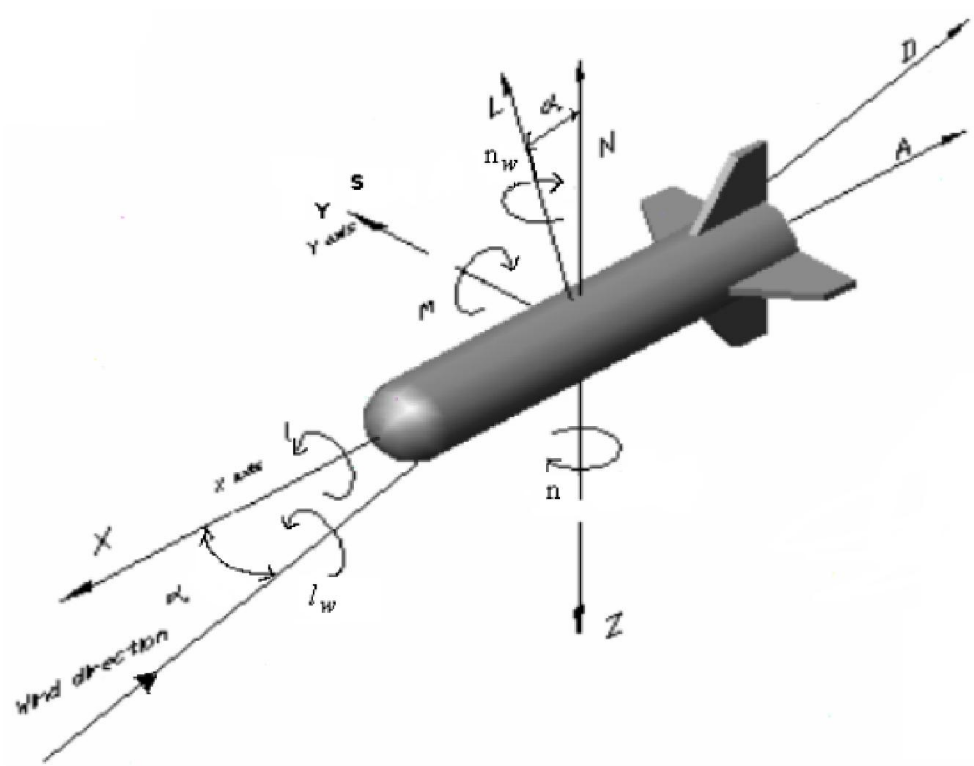


Figure 2: Axes, forces and moments sign convention for body and wind axes

<u>Forces in wind axes</u>:

Drag force, D	:	along the wind direction
Side force, S	:	to starboard
Lift force, L	:	Normal to wind direction

<u>Moments in wind axes</u>:

Rolling moment, l_w	:	starboard wing down
Pitching moment, m_w	:	nose up
Yawing moment, n_w	:	nose towards starboard

Therefore, the relations of forces at balance, body and wind axes with right handaxes convention are as follow

$$\vec{F}_b \begin{bmatrix} F_{Xb} \\ F_{Yb} \\ F_{Zb} \end{bmatrix} = \begin{bmatrix} -A_{xb} \\ Y_b \\ N_b \end{bmatrix}, \quad \vec{F}_B = \begin{bmatrix} F_{XB} \\ F_{YB} \\ F_{ZB} \end{bmatrix} = \begin{bmatrix} -A \\ Y \\ -N \end{bmatrix} \text{ and } \vec{F}_w = \begin{bmatrix} F_{Xw} \\ F_{Yw} \\ F_{Zw} \end{bmatrix} = \begin{bmatrix} -D \\ S \\ L \end{bmatrix} \qquad (1)$$

Subscript 'b' stands for balance axes, 'B' stands for body axes and 'w' for wind axes

Moment components are consistent with the positive convention of the axes. Thus, equations for moments in the balance, body and wind axes will be

$$\vec{M}_b = \begin{bmatrix} M_{Xb} \\ M_{Yb} \\ M_{Zb} \end{bmatrix} = \begin{bmatrix} Mr_b \\ Mp_b \\ My_b \end{bmatrix}, \quad \vec{M}_m = \begin{bmatrix} M_{XB} \\ M_{YB} \\ M_{ZB} \end{bmatrix} = \begin{bmatrix} l \\ m \\ n \end{bmatrix} \text{ and } \vec{M}_w = \begin{bmatrix} M_{Xw} \\ M_{Yw} \\ M_{Zw} \end{bmatrix} = \begin{bmatrix} l_w \\ m_w \\ n_w \end{bmatrix} \qquad (2)$$

The angles are measured w.r.t wind axis and clockwise rotation from reference axis to the target axis is considered positive.

Data Analysis

The balance data analysis involves calculation of aerodynamic coefficients in body as well as in wind axis. These analyses involve transformation from one coordinate systemto another, with origin of the two systems involved in transformation to be coinciding andalso, transfer of forces and moments from a point in the reference axes to another point inthe target axes.

The balance data analysis sequence thus, involves calculation of balance components using the balance calibration coefficient matrix 'Cij' and balance signal. The equation involved is

$$\vec{F}_i = C_{ij}\vec{E}_j$$

where, $F_i = \{Ax, N_1, N_2, S_1, S_2, Rm\}$ and $E_j = \{E_{Ax}, E_{N1}, E_{N2}, E_{S1}, E_{S2}, E_{Rm}\}$

The forces and moments are evaluated at the balance center using equations given below

$$A_{xbc} = A_x \qquad Rm_{bc} = Rm$$

$$S_{bc} = S_1 + S_2 \qquad Mp_{bc} = (N_1 - N_2) \times d$$

$$N_{bc} = N_1 + N_2 \qquad My_{bc} = (S_1 - S_2) \times d$$

where, d is the distance from balance center to normal force or side force gauges.
From the balance center, transformation is done to body or model axes, aligning the force arms of the two axes using following equation

$$\vec{F}_B = T_b^B \vec{F}_b \tag{3}$$

$$\vec{M}_B = T_b^B \vec{M}_b \tag{4}$$

where, $\vec{F}_B$ and $\vec{F}_b$ denote force vectors and the suffix indicates the model and balance axes system respectively. Similarly, $\vec{M}_B$ and $\vec{M}_b$ denote moment vectors. 'T_b^B' is the transformation matrix given as

$$T_b^B = \begin{bmatrix} \cos\alpha_o \cos\psi_o & \cos\alpha_o \sin\psi_o & -\sin\alpha_o \\ (\sin\phi_o \sin\alpha_o \sin\psi_o - \cos\phi_o \sin\psi_o) & (\sin\phi_o \sin\alpha_o \sin\psi_o + \cos\phi_o \cos\psi_o) & (\sin\phi_o \cos\alpha_o) \\ (\cos\phi_o \sin\alpha_o \cos\psi_o + \sin\phi_o \sin\psi_o) & (\cos\phi_o \sin\alpha_o \sin\psi_o - \sin\phi_o \cos\psi_o) & (\cos\phi_o \cos\alpha_o) \end{bmatrix}$$

Here the angles are the offsets between the balance and the body.
The forces and moments are then transferred to a reference point (e.g. CG of themodel) in the body axes. During the transformation, forces remain same, whereas themoments are changed due to displacement vector. The following equations are used forthe moment calculation.
If point 1 is balance center and point 2 is the reference point in model and a systemof forces produces a resultant force $\overline{\mathrm{F1}}$ and a resultant moment $\overline{\mathrm{M1}}$ relative to point 1 then the equivalent system acting at another point 2 is

$$\vec{F}_2 = \vec{F}_1$$

$$\vec{M}_2 = \vec{M}_1 - \vec{r}_{12} \times \vec{F}_1$$

Thus,

$$\vec{M}_2 = (iM_{x1} + jM_{y1} + kM_{z1}) - \vec{r}_{12} \times \vec{F}_1$$

$$= \hat{i}\left[M_{x1} - (\Delta y F_z - \Delta z F_y)\right] + \hat{j}\left[M_{y1} - (\Delta z F_x - \Delta x F_z)\right] + \hat{k}\left[M_{z1} - (\Delta x F_y - \Delta y F_x)\right]$$

Therefore, the final equations of the moment about the three axes for model are

$$l = M_{xB} = M_{xb} - (\Delta y F_{zb} - \Delta z F_{yb})$$

$$m = M_{yB} = M_{yb} - (\Delta z F_{xb} - \Delta x F_{zb})$$

$$n = M_{zB} = M_{zb} - (\Delta x F_{yb} - \Delta y F_{xb})$$

Thus, from equation 3 and 4, forces and moments are calculated and coefficients areevaluated at the model reference point. Equations for force coefficients are

$$C_A = \frac{A}{qA},\ C_Y = \frac{Y}{qA} \text{ and } C_N = \frac{N}{qA}$$

And for moments coefficient

$$C_l = \frac{l}{qAs},\ C_m = \frac{m}{qAs} \text{ and } C_n = \frac{n}{qAc}$$

Suggested works:

- Draw the schematic of six component force balance calibration setup and describe the calibration procedure in details.
- Write the equations for evaluating the orthogonal forces and moments at the force balance center acted upon by a random force.
- Write the equations for real orthogonal forces and moments acting on a model (attached to the force balance) at a reference point by a random aerodynamic force in terms of loads measured by the force balance.
- Discuss what you have understood from this experiment.
- Calculate the dimensional less , drag , and moment coefficient

Questions:

1) What are the different aerodynamic forces acting on a model?
2) What are the working principles of this force balance?
3) What are the other types of balances?
4) What are the advantages and disadvantages of mechanical balance compared to strain gage type?
5) Why we do no wind analysis in force measurement experiment.

References:

1) Low-speed wind tunnel testing - W. E. Rae, Jr. and Alan Pope.
2) Fundamental of Aerodynamics - J. D. Anderson.
3) Wind-tunnel technique - R. C. Pankhurst and D. W. Holder
4) Wind-tunnel testing - Alan Pope

5) Low-speed wind tunnel testing - W. E. Rae, Jr. and Alan Pope
6) High-speed wind tunnel testing - Alan Pope and K. N. Goin
7) Aerodynamics test facilities - T. N. Krishnaswamy
8) Fundamental of Aerodynamics - J. D. Anderson
9) Boundary layer theory - H. Schlichting

Experiment # 4

Studies on Subsonic jet propagation

Learning Objective:

To study the development of velocity profiles in an axisymmetric jet by using a single Pitot tube with traverse mechanism.

Theory

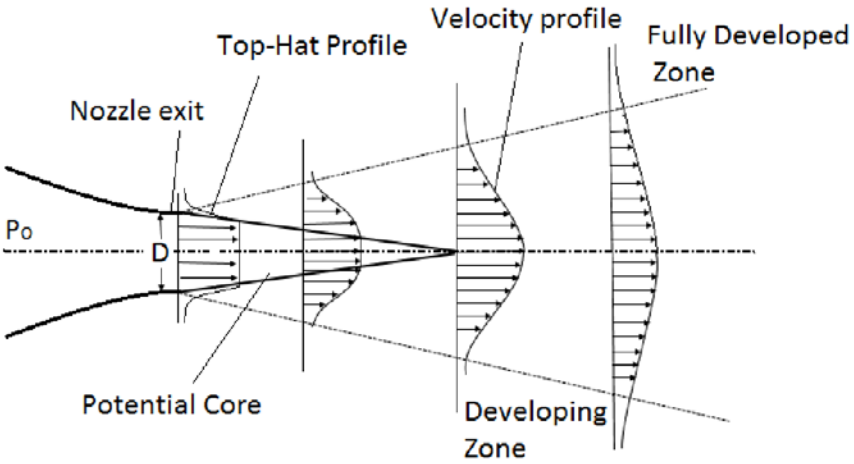
Definition

A jet may be defined as a pressure driven shear flow that exhibits a characteristic that the ratio of its local width to axial distance is a constant. This constant assumes a value of 8 for jet Mach numbers less than 0.2, and the constant decreases with increase in Mach number. The jet may also be defined as fluid flow issuing from a nozzle into a medium of lower speed fluid or stagnant atmosphere. As the jet fluid travels further away from its origin, it slows down due to mixing with stagnant ambient fluid entrained and inducted into the jet field. This is due to the boundary layer at the nozzle exit which develops roll-up structures or ring vortices that grow in size as they move downstream, due to continuous entrainment of the ambient fluid into the jet stream [1]. Thus, mass flow at any cross-section of the jet progressively increases along the downstream direction. Hence, to conserve momentum, the centerline velocity decreases with downstream distance. The differential shear at the jet boundary forms vortices, and they bring fluid mass from the surrounding environment into the jet field. This transport of mass from the surroundings into the jet is called *entrainment*. The large scale vortices are mass entrainers while the small scale ones are good mixing promoters.

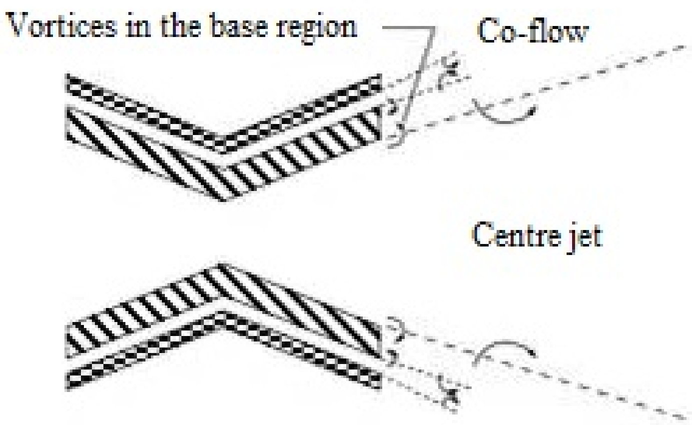
Classification of jets

a) Based on configuration

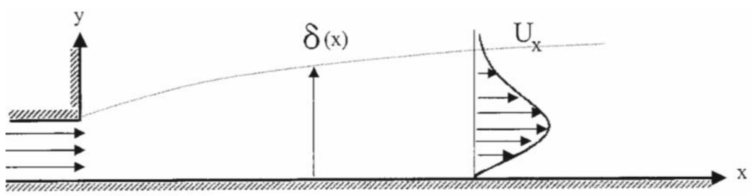
If jet is issued in stagnant atmosphere, it is termed *submerged jet*. If the jet is surrounded by a fluid of finite velocity, then it is termed *co-flowing jet*. It can be further classified into subsonic and supersonic co-flow jet depending on the Mach number of the surrounding fluid. The term *impinging jet* is used when jet is delivered normal to a solid or deformable boundary. The combination of jets normal to each other is termed *opposing jets*. When the jet impinges on a solid boundary such that the jet axis meets it obliquely, the jet is termed *wall jet.* The types of jets are illustrated in figure 1.



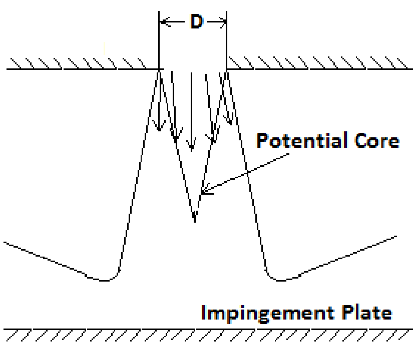
(a) Submerged jet



(b) Co-flow jet



(c) Wall jet [2]



(d) Impinging jet [3]

Figure 1 Types of jet

b) Based on Mach number and compressibility

Jets with Mach number less than 1 are called subsonic jets. The compressibility effects can be neglected for Mach number less than 0.3 hence these are called incompressible subsonic jets. On the other hand, compressibility becomes significant beyond Mach 0.3 hence jets in this regime are termed as compressible subsonic jets. If the jet Mach number exceeds 1 then they are termed as supersonic jets. It is obvious that all supersonic jets are compressible. Jet classification is shown in figure 2.

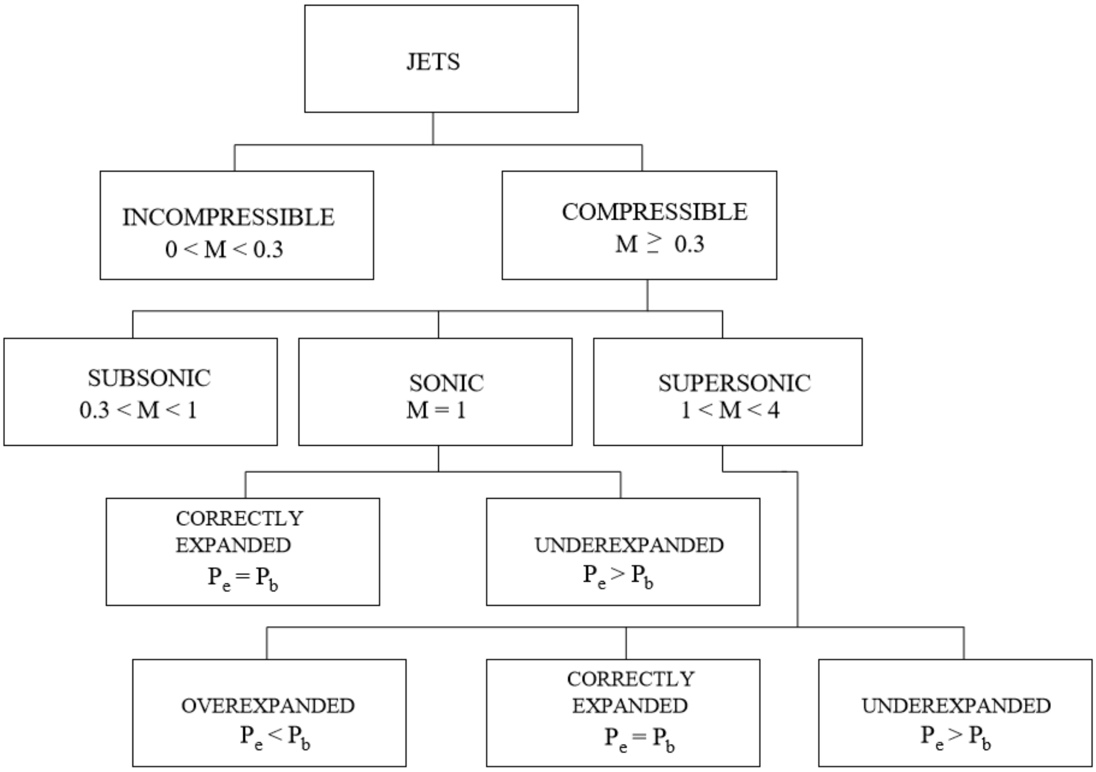


Figure 2 Classification of jets [1]

Subsonic jets

Subsonic jets have a distinct characteristic of always being correctly expanded. Different regimes of subsonic jets are the following.

Potential core region – It is defined as the extent up to which the axial velocity remains preserved and is equal to the jet exit velocity. The core is annularly surrounded by developing shear dominated mixing layer which has intense turbulence. It is due to the shear effects that make all jets turbulent and the theoretical critical Reynolds number tends to 0. Experiments suggest that the potential core region extends axially up to 6 times the nozzle diameter D, primary reason being that the mixing initiated at the jet boundary has not yet permeated enough to affect the axis velocity.

Transition region – In this region, decay in the centerline axis velocity is taking place. This region extends from 5D to 10D. High mass and momentum transport owing to turbulence smoothens the velocity profile and decreases the velocity gradient between the jet and the ambience.

Fully developed region – This is the region of self-similarity where the jet becomes similar in appearance to a flow of fluid from a source of infinitely small thickness. The jet velocity loses its significance after 30D. All the regions of a subsonic jet is depicted in figure 3.

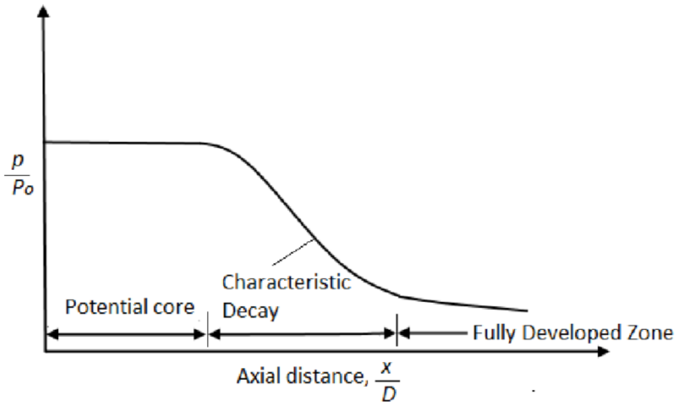


Figure 3 Subsonic jet development [3]

c) Based on level of expansion at the nozzle exit

The terminology nozzle pressure ratio (NPR) refers to the ratio of stagnation pressure of the settling chamber p_0, which runs the jet delivering nozzle, to the pressure of the ambient p_a to which the jet is discharged. The NPR is the primary governing parameter for the level of expansion at the nozzle exit.

Overexpanded jet - A jet is called overexpanded when the pressure at the nozzle exit p_e is less than the ambient pressure p_a into which it is being discharged.

Correctly expanded jet - A jet is known as correctly expanded if the pressure at the nozzle exit is same as that of the ambience.

Underexpanded jet - Under expansion of jet prevails when the nozzle exit pressure is more than the ambient pressure where the jet is being delivered.

Applications

a) Combustors

An important application of supersonic mixing enhancement is in scramjet combustion. While the axial thrust consideration based on the momentum of the injected fuel makes parallel fuel injection desirable, high speed parallel or near parallel shear flows spread very slowly. This results in long combustors to complete the fuel-air mixing. However, because of the drag and weight consideration at high speed, a relatively short combustor is highly desirable. Thus, the inherently low flow mixing rate associated with supersonic shear layers must be increased by employing mixing enhancement techniques.

b) Noise reduction

Jet noise reduction has been a topic of research since the first commercial jet flights. Most of the reductions in noise have been achieved by lowering jet exit velocities. Additional noise mitigation has been achieved through nozzle exit geometry modifications by using grooves and tabs at the nozzle exit. However, these designs are largely chosen through a cut-and-try technique whereby many designs are tested and the best is chosen for use.

c) Base heat reduction of launch vehicles

The base pressure field in the case of rocket and missile plays a vital role from drag point of view and also due to the fact that the high temperature plumes issuing from the nozzle will get attracted towards the base region when the base pressure is at sub-atmospheric level. This can cause significant base heating. The heated base might affect the electronic devices which are housed inside the rocket shell. Therefore, it is essential to make sure that the hot exhaust gas coming out of the nozzle is mixed with the cold ambient air, as efficiently as possible. This mixing will result in reduced temperature of the jet stream, leading to reduction in base heating.

d) Metal deposition

The area of metal deposition and metal coating using liquid dynamic compression may also benefit from supersonic mixing studies. To produce fine powders of reactive metals such as aluminium, magnesium and titanium with uniform size, a supersonic coaxial jet of inert gas can be discharged over a laminar centre jet containing the molten metal. The atomization of metal occurs in two-phase shear flow where the high shear and rapid cooling from supersonic expansion result in fine crystal structure and low level metal oxidation.

e) Thrust vector control

Thrust vector control of aircraft is commonly obtained with the use of either fluid dynamic means such as secondary asymmetric injection of gas or liquid or mechanical means such as physically moving the nozzle exhaust bell and tabs. Typical problems associated with these methods are the added weight of the injectant and control surfaces, the integrity of movable control surfaces at hostile flow conditions, and the thrust loss associated with the flow blockage.

f) Gas dynamic laser

Hot gases expand through appropriately shaped nozzles from a high-pressure and temperature chamber into a low pressure chamber. This creates a highly non-equilibrium region were a strong population inversion takes place. Exploiting this environment very high laser output power can be achieved. The diffuser is used to slow down the supersonic flow to subsonic speeds, and then the gases are generally exhausted to the atmosphere. High power CO_2 lasers are used for drilling, cutting, welding, heat-treating and alloying in the manufacturing industries. Some of these lasers are so powerful that the beams emerging from the chamber could potentially destroy the optics.

g) Air knives

An air-knife is a fancy name for using air to blow a substance from a surface. A crude example can be seen in many machine shops. A machine operator will blow metal chips and oil from a newly machined part using a nozzle attached to a high-pressure (say, 100 psi) air line. The same effect can be achieved by using a properly designed nozzle using low-pressure air to remove water, oil, dust, and even more viscous contaminants from a surface. Most air-knives use low-pressure air because it is much less expensive to produce, much safer for workers, and just as being more effective than high-pressure air. High-pressure air is used only for applications where lower pressures do not produce enough force. Typically, air-knives are used on conveyor systems to clean continuously produced parts. For example, air-knives might be used to remove machining chips from aluminium extrusions before anodizing or painting as the extrusions are carried past on a conveyor belt. They might be used after painting to decrease drying time by blowing air onto the parts as they pass by on a hook-chain conveyor. Air-knives can also be used to clean open web conveyor belts to prevent product or debris from clogging the openings. Scrapper knives are sometimes used to remove stickies from conveyor belts. The knife scrapes the stickies of the belt and blows them into a bin. Air-knives can be used to reduce friction by blowing air beneath paper or steel sheets, thus providing an air cushion on which they can ride. Also, air knives are often used as a barrier between two areas to prevent mixing of air. For example, where doors are not practical, blowers with nozzles can be used to blow air across a doorway, such as a fork truck entrance, to prevent hot or cold outside air from mixing with conditioned air. As parts are lifted from a washing or plating bath, cleaning or plating solution adheres to the parts. This is called drag out. The wash solution can cause contamination of downstream processes or rinse tanks. Air-knives can be used to blow solution from the parts back into the process tank as they are lifted from the tank. This reduces the need for rinsing further downstream and reduces contamination of downstream processes.

h) Water jet cutting

Water jet cutting is a growing technology nowadays which is used worldwide for cutting metals with very high efficiency. A high speed water jet is made to impinge on the metal surface, and the sheer momentum of the jet makes it to penetrate the metal with ease. This is one of the practical applications where we desire to inhibit mixing and increase the core length of the jet.

i) Gas welding/cutting

This is yet another example of case where we desire that the core of jet, be preserved. Elongated core length helps in better results as the high temperature of the flame penetrates deeper into the metal.

Experimental Setup

The experiments are to be conducted in the open jet facility at High Speed Aerodynamics Laboratory, Indian Institute of Technology Kanpur. The test facility consists of compressors, storage tanks and an open jet test facility along with a six degree of freedom traverse system, for taking the measurements. A schematic diagram of the open jet facility laboratory is shown in figure 4.

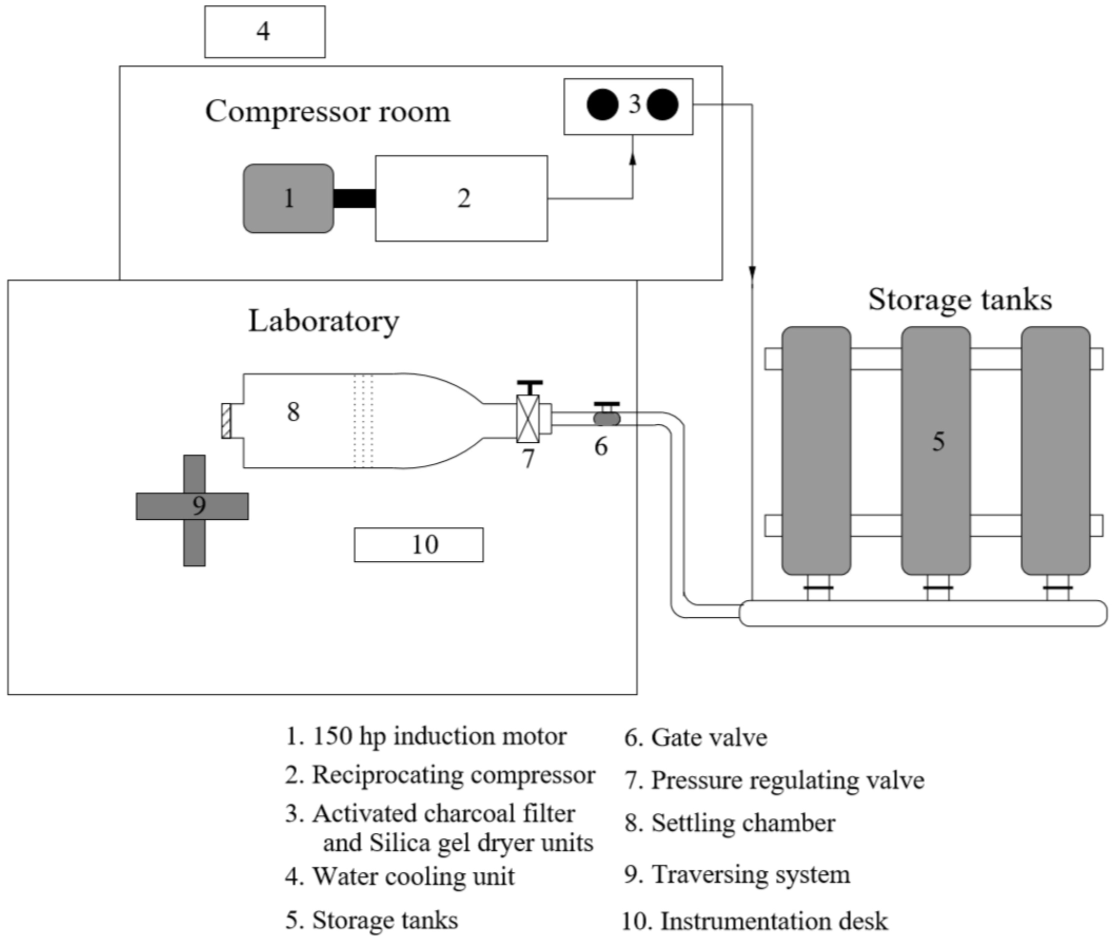


Figure 4 Schematic diagram of the open jet test facility

a) Compressed Air Supply System

The compressed air supply system consists of a two stage reciprocating compressor, competent of delivering volume flow rate of 360 ft^3/min of air at a pressure of 500 psi. The compressor is driven by a 150 hp, 3-phase induction motor. The compressed air is subsequently cooled by passing it through an inter-cooler and then supplied to a pre-filter consisting of porous stone candles, to remove minute solid contaminants and oil droplets. An activated carbon filter is employed for finer filtering of the flow. The compressed air is dried in a dual-tower semi-automatic silica gel dryer. A portion of the dried air is heated and is used to alternately re-activate each tower. A diaphragm-type pressure valve, operated by a pressure relief pilot, permits the air dryer to operate at 500 psi as the pressure in the receivers builds up from atmospheric to the required stagnation pressure. The air is stored in three tanks, which has a total capacity of 3000 ft^3. The jet facility control section includes a gate valve followed by a pressure regulating valve. The gate valve is of quick acting type which enables the air to pass through a particular pipe, whereas the regulating valve is more sensitive to the control action and is used to maintain the desired value of pressure in the stagnation chamber. The pressure regulating valve is connected to a 1 m long mixing tube, of 3 inches diameter, and then to a settling chamber. This intermediate mixing tube enables the flow to develop from the valve exit to the entrance of the stagnation chamber.

b) Open Jet Test Facility

The open jet facility consists of a cylindrical settling chamber attached to the high pressure storage tanks. Figure 5 depicts a schematic diagram of the jet test facility. The settling chamber is connected to the mixing tube by a wide angled diffuser followed by three screens placed consecutively. These screens have finely meshed wire grids, set 3 cm apart from each other, for minimizing the turbulence level at the nozzle inlet. These screens are inserted in the settling chamber where the flow is almost at rest and the air is theoretically at 0 Mach number. The settling chamber is 600 mm in length and has a constant area circular cross-section of 300 mm internal diameter. The settling chamber has provisions in the form of tapings, for stagnation pressure and temperature acquisition. The test models are set at the end of the settling chamber by a slot holder arrangement, which is a short pipe like protrusion with embedded O-ring to prevent air leakage. Model to be studied is placed over the O-ring, over which an annular retaining sleeve with internal threads is screwed tightly.

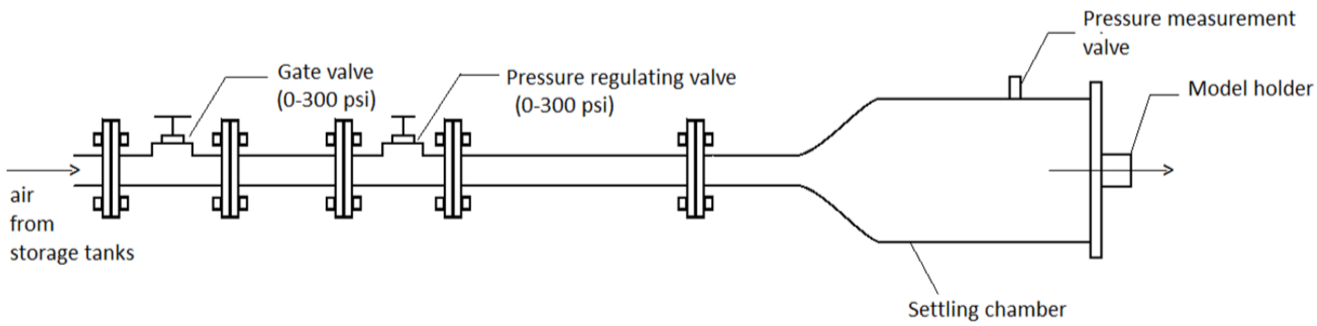


Figure 5 Schematic diagram of a free jet test facility

The settling chamber total pressure (p_0), which is the governing parameter in this experiment, is to be maintained constant with an accuracy of $\pm$0.1%, during all runs by controlling the pressure regulating valve. The stagnation pressure (p_0) in the settling chamber, gives various nozzle pressure ratios (NPR), defined as the ratio of stagnation pressure to the back pressure (p_0/p_b) required for any study. The settling chamber total temperature is the same as the atmospheric temperature and the back pressure is the ambient pressure into which the jet is being issued. The settling chamber pressure is to be measured by a pressure transducer, during each of the experimental run, while the mercury barometer placed in the laboratory, averaged over the duration of experiment, measured the day-to-day changes in ambient pressure (p_a). A scientific thermometer is to be used to measure the ambient room temperature.

Instrumentation for Pressure Measurement

a) Pressure Probe

The pressure-sensing probe employed is a conventional pitot probe. The pitot probe has been one of the most useful and frequently used pressure measuring instruments in experimental fluid dynamics. The accuracy of the pitot probe depends on various factors which include the Reynolds number, its shape, its orientation with respect to the mean flow direction, turbulence intensity, length scale, magnitude of traverse shear and finally the Mach number.

Schematic view of a pitot probe and its attachments can be seen in Figure 6. The pitot probe has the internal and external diameters of 0.4 mm and 0.6 mm respectively. The values of Re are to be kept much higher than 500, which adversely affects the pressure measurements by the probe. Owing to this, we can neglect the viscous effects on pitot pressure measurements, and consider them insignificant. The angularity, induced in the flow due to the probe is within the range of ±20° hence any error due to angularities can be neglected [1].

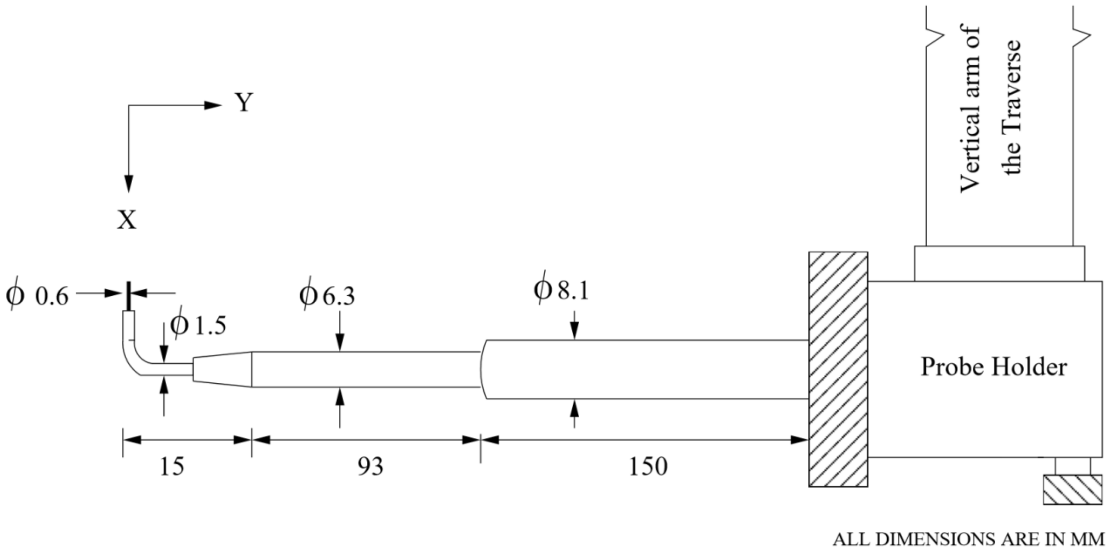
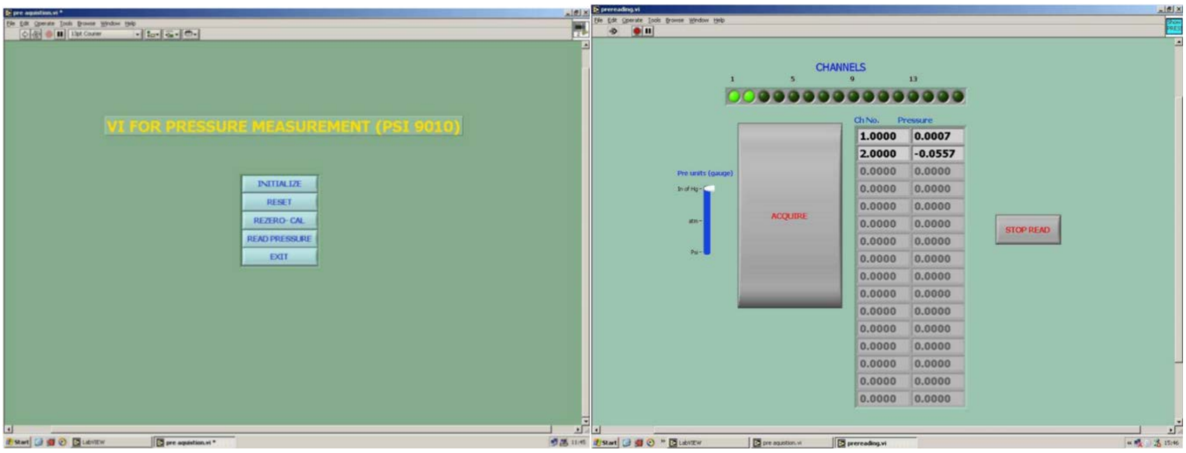


Figure 6 Schematic diagram of the pitot probe

Throughout the pressure measurements, the sensing probe stem is oriented parallel to y-axis with the sensing probe facing the jet axis (x-axis). The pitot probe is mounted on a three-dimensional traverse having six degrees of freedom, also including a probe-yawing mechanism. The traverse has linear resolution of 0.1 mm along all the three traverse axes (x, y and z) hence; the positioning accuracy of the probe is within $\pm$0.1 mm.

b) Pressure Transducer and Application Software

The pitot pressure sensed by the probe is measured using a *Pressure Systems model 9116*, 16-channel pressure transducer (interfaced with a Pentium 4 computer installed with VI based software for data acquisition and analysis). The model 9116 transducer is capable of measuring pressures up to 300 psi, which is approximately 20 atm. The transducer also has a facility to choose the number of samples to be averaged, by means of dip-switch settings; for the current experiment 250 samples are to be chosen for averaging. The accuracy of the transducer (after re-zero calibration) is specified to be within ± 0.15% of the full scale. Also, transducer offset errors are eliminated by performing a re-zero calibration preceding every run.



(a) Main front panel (b) Pressure acquiring front panel

Figure 7 VI for pressure acquisition

The system 9116 intelligent pressure scanner is a pressure measuring device intended for use in test and production environments. It consists of 16 channels and operates in deferential mode. It has an asynchronous RS 422/485 host communication interface. It also has a standard RS-232 diagnostic interface that can also be used as host interface. The optomux style command set is used to send commands and receive response from all ports. It may be configured to communicate in the multi-drop network communication, always at a selected baud rate, using the optomux protocol. The multi-drop communication always operates with no parity, 8 data bits and 1 stop bit. The default baud rate is set to 9600. Changes to the baud rate can be made using special procedure via the DIP switch used to select the node address during initialization at power up. During this special baud rate selection procedure, the number of averages used during the data acquisition may also be selected.

The application software developed using the LabVIEW 8.2 (shown in Figure 7) links the host computer to the pressure scanner via RS 232 communication. The application software performs all the required functions like initialization, reset, re-zero calibration and reading pressure.

Experimental Procedure

a) Centerline Pitot Pressure Distribution

The centerline pitot pressure distribution for subsonic jet are to be measured by placing the pitot probe along the jet axis (x-axis), and traversing from the exit plane of the nozzle in the downstream direction. The pitot probe is moved along the jet axis, at intervals of 1 mm, up to 32D.

b) Pitot Pressure Profiles

The pitot pressure measurements are to be taken along the lateral direction (Y-axis) for plotting radial pressure profiles. Here, the pitot pressure (p_t) measured is divided by the stagnation pressure of the settling chamber (p_0) for nondimentionalizing, and plotted against the non-dimensionalized lateral distances, y/D and z/D.

Data Accuracy

The possible sources of error may be due to the following factors:

- The linear movement of traverse along x, y and z directions.
- The settling chamber stagnation pressure measurement.
- The error in the measurement of total pressure in the jet field.
- The possible inaccuracies in nozzle dimensions.

Precautions to be Observed

In addition to the measures taken to minimize errors, like linearity checks, re-zero calibrations, etc., the following precautions were also observed during the experiments:

- The horizontal alignment of the settling chamber is to be ensured.
- Wire-mesh screens are placed inside the settling chamber to reduce the disturbances and lower the turbulence level at the nozzle inlet.
- The pitot probe is to be cautiously aligned with the flow direction to the x-axis of the nozzle geometry, facing the nozzle exit.
- Care is to be exercised while aligning the models, to ensure proper positioning of the measurement planes.
- The pressure pipelines and the settling chamber ports should be ensured to be leak free.
- During the experiment, the stagnation pressure reading is to be constantly monitored.
- The stagnation pressure and the air flow rate in the settling chamber are kept constant during the experiments by continuously adjusting the pressure regulating valve.

Important relations:

$$\int_{-\infty}^{+\infty} u^2 dy = const. = \int_{-\infty}^{+\infty} \tilde{u}^2(y) dy = J/\rho$$

J=Momentum flux in the jet per unit span

$$\dot{m} = \left(36J\rho^2 \nu x\right)^{1/3}$$

m=mass flux

Typical centerline pitot pressure decay

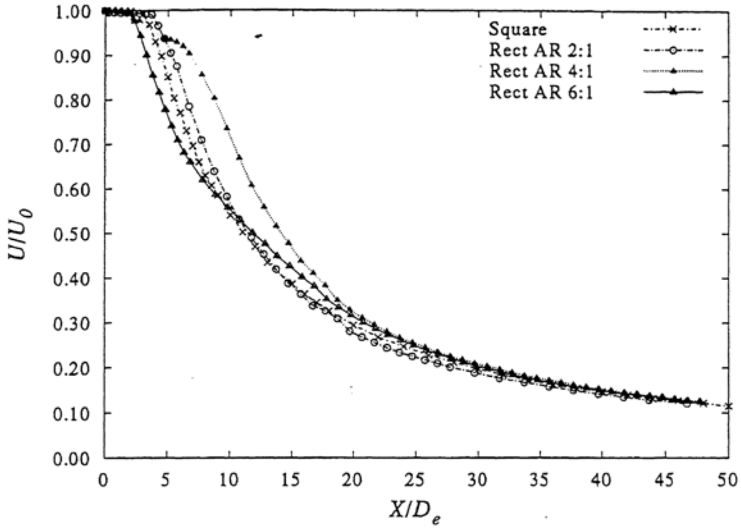
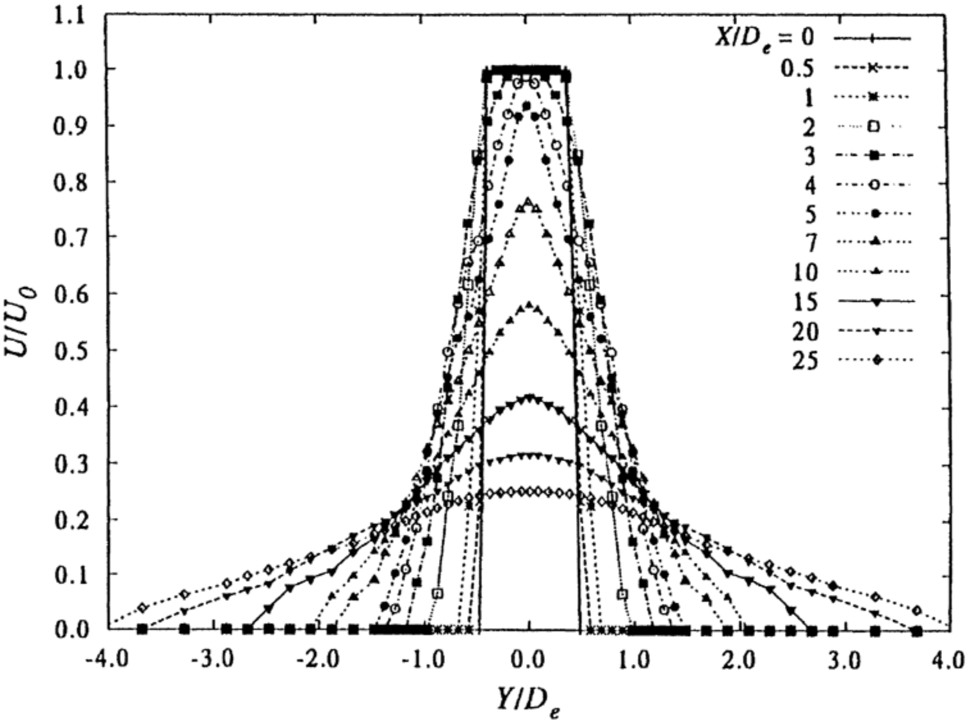


Figure 4.8 Centreline velocity decay of various rectangular jets at Mach number 1.0

Typical jet profile



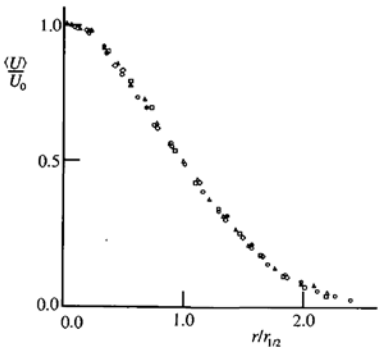


Fig. 5.3. Mean axial velocity against radial distance in a turbulent round jet, Re $\approx 10^5$; measurements of Wygnanski and Fiedler (1969). Symbols: ○, $x/d = 40$; △, $x/d = 50$; □, $x/d = 60$; ◇, $x/d = 75$; •, $x/d = 97.5$.

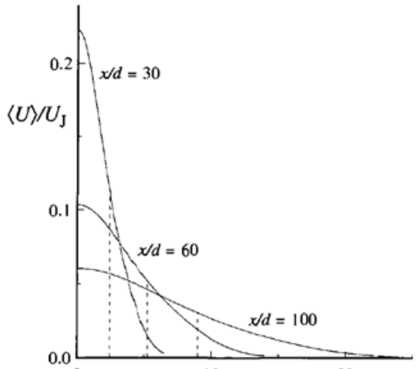


Fig. 5.2. Radial profiles of mean axial velocity in a turbulent round jet, Re = 95,500. The dashed lines indicate the half-width, $r_{1/2}(x)$, of the profiles. (Adapted from the data of Hussein *et al.* (1994).)

Proposed Plan:

a) Draw and explain the schematic sketch of the jet flow system and how to carry out the experiments in this facility.

b) Measure the centerline velocity in the jet at x/d (0 to 35) locations. Plot nondimentionalized pressure and Mach no. decay with respect to downstream distance.

c) Measure and plot the velocity profiles of a jet at x/d = 0,10, 20,30,32,34 with M = __(will be mentioned during lab)

d) From the acquired data, compute jet half width, mass flux, and momentum per unit span at these locations.

e) Show that in the near field the jet is developing and in the far wake jet has developed fully. Also show the variation of mass flux with downstream distance and show that it increases as more and more mass entrains in.

f) Plot momentum flux per unit span and show that it is constant with downstream distance.

Questions:

1) What is invariant across a jet?

2) Give two practical applications for jet mixing.

3) What is jet width? How is it calculated from the velocity profile?

References:

1. Fluid Mechanics - Kundu and Cohen

2. Turbulent flows – S.B.Pope

3. Flow and noise characteristics of non-circular jets Ph.D Thesis K.Srinivasan (A128598)